ANALYSIS AND DESIGN OF MULTI-STOREY (G+5) FRAMED STRUCTURE USING STAAD PRO

Krishna Kumar Kumbhkar[1], Ahsan Rabbani[2]
[1]Post graduate student, Department of Civil Engineering, Kalinga University,
New Raipur, Chhattisgarh, India

[2]Assistant Professor, Department of Civil Engineering, Kalinga University,
New Raipur, Chhattisgarh, India

Abstract: *This Paper is the overview of the work done for the design and analysis of the multi-storey building (G+5) under the effect of various forces acting on the building such as dead load, imposed load, wind load and seismic load. These forces acting on the building shows that if the structures are not well designed and constructed with adequate strength this may leads to the partial or complete collapse of the multi-storey structures. To ensure the safety against these various forces acting on multi-storey building, there is need to analyse and design the structures so that the structure can withstand these forces. The main objective of this work is to study the wind and seismic performance on the structures. Here in this work residential building of G+5 storied structure considered for the analysis purpose which is located in Raipur city the capital of Chhattisgarh state which comes under the zone II for the analysis of seismic load. Any structure is subjected to effects of various forces due to dead load, live load, wind forces and seismic forces throughout its life span. Static load is due to dead load and imposed load and dynamic load is due to wind load and seismic load. The whole structure analysed by the help of STAAD PRO software.*

Keywords: Multi-storey; Load; STAAD PRO; Collapse; Seismic analysis; Structure

1 INTRODUCTION

In the rapid growth of the world, population explosion and lack of ground area for the residential purpose near urban and metro cities of India needs to more and more construction of residential and commercial building for the infrastructural development. In the early age of 19th century there were no requirement of construction of multi-storey structure and they adopt horizontal system of construction because of less population and lack of infrastructural development. But with the growth and development of human civilization in 21st century infrastructural development is its peak. Hence the vertical system of construction is adopted to accommodate the urban population of the country since there is less availability of ground area in cities. A multi-storey building is a structure with number of vertical floors and number of bay in transverse and longitudinal direction of the structure. A structure is design and develop to fulfil the requirement for which it is designed. A structure is subjected to various forces throughout its whole life span such as forces due to dead load, live load, wind load and seismic load. For the analysis and designing purpose of multi-storey structure it is necessary to estimate these forces so that the structure will be safe during its service life. In this project work (G+5) multi-storey structure is considered with each floor having height of 3m. In this structure rectangular beam and column is used for designing the framed structure. The size of the beam is taken as 500mm*300mm and size of column is considered as 450mm*450mm. The structure is analyze for the dead load, live load, wind load and seismic load and result is observed in structural term of axial force, shear force and bending moment. The building is subjected to both horizontal loads and vertical loads. Vertical loads are due to the dead load of structure and live load acting on the structure and horizontal loads are due to wind load or pressure intensity of wind on the structure and seismic load which depends on the seismic zone. In this project for seismic analysis, seismic zone II is considered for Raipur city. Analysis and design of (G+5) multi-storey structure is done by a world's leading and powerful structural analysis and designing software STAAD PRO.
The basic steps to carry out the design and analysis is:

(a) Model generation
(b) Application of loads and Calculation
(c) Result analysis
(d) Structural member design

The STAAD PRO software provides graphical user interface to generate the model, which can be analyzed by using the STAAD engine. After the analysis and design the result can be seen graphically. STAAD PRO uses general purpose calculation engine for analysis and concrete design. STAAD PRO consist of International design codes so it is convenient to design the structure by using any international code. The structure is designed for dead load, live load, wind load and seismic load as per **IS 875** for dead, live and wind loads (Part I, II, III) and **IS 1893-2002** for seismic load.
The objective of this research work is to first create a geometric model of the structure, to apply various load and load combination on the structural member, to analyze the result obtained, and last to design the structural members by using STAAD PRO.

2. LITERATURE REVIEW

A 30 storey high rise structure is analyzed for wind load and seismic load on a project report by [1]. They uses the structural analysis software STAAD PRO 2008 to achieve their project goal. They found that more reinforcement is required for static analysis as compare to dynamic analysis. Also concluded deflection and shear bending is more in dynamic analysis as compare to static analysis. Lower beam requires more steel for dynamic load than static load.
Project study on G+28 tall structure analyzed by using the software STAAD PRO by [2]. Combination of static and wind load is considered and result is analyzed in term of axial force, shear force, bending moment and support reaction. Story wise variation in result with respect to structural parameters are compared.
A project report on analysis of a multi-storey framed structure of G+6 is carried out by [3]. The structure is consist of apartments in each floor. Dead load and live load is applied and obtained the design of beam, column and footing. They found the software STAAD PRO time saving and accurate for the calculation of cumbersome data of the structure.
Project is prepared about the static and dynamic analysis of seismic performance of G+15 multi-storey structure [4]. Analysis is done for the ordinary moment resisting frame and special moment resisting frame by using the software STAAD PRO. Analysis is performed for the seismic zone II. It is observed that special moment resisting frame is better than ordinary moment resisting frame in resisting seismic forces.

3. METHODOLOGY

While designing the high rise structure safety and serviceability of the structure is an important consideration for an engineer. The various loads acting on it should be properly estimated and on the basis of these data design is prepared along with this, it should fulfill all the expectation of the customer or client. For the designing of (G+5) multi-storey structure the limit state method of design is adopted as per the **IS 456-2000**. STAAD PRO uses Indian standard code of practice and limit state method of design for concrete design. The design steps are:

(a) Geometrical modeling
(b) Application of load
(c) Analysis
(d) Designing

4. LOAD AND LOAD COMBINATION

Various load acting on the structure considered for the analysis purpose and the result obtained shown with the help of figure. Loads such as dead load, live load, wind load, and earthquake load considered in the analysis work. Load combination also considered.

Dead load (DL): Dead load is the first vertical load of the structure. Dead load is permanent or stationary load of the structure which consist of load of column, beam, roof slab, wall and these loads are transferred to the underneath soil through foundation. Dead load of the structure is calculated as per **IS 875** part I.

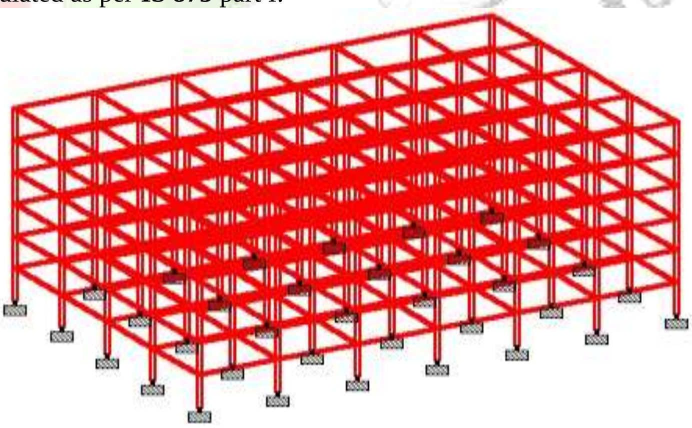

Fig 1. Shows structure after application of dead load

Live load (LL): Live load is the second vertical load of the structure. Live load is movable or moving load without any acceleration or impact. Live load is time dependent load which changes time to time. The minimum value of live load can be assumed as per the **IS 875** part II.

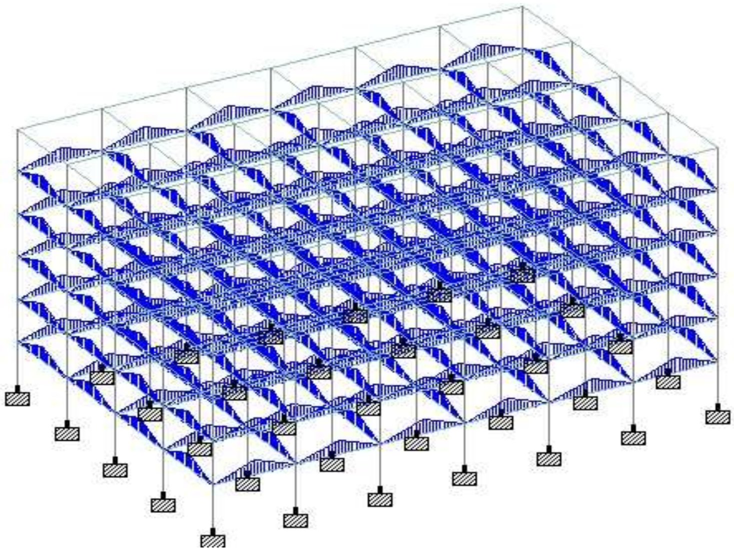

Fig 2. Shows structure after application of live load

Wind load (WL): Wind load is considered as the first horizontal load on the structure which is primarily due to the movement of air relative to the earth. Wind load is considered during the design of high rise structure. For Raipur city basic wind velocity V_b = 39 m/sec. is selected as per the IS 875 part III. To calculate the design wind velocity V_z following expression can be used.

$$V_z = k_1 * k_2 * k_3 * V_b$$

Where k_1 = Risk coefficient
k_2 = Coefficient based on terrain
k_3 = Topography factor
V_b = Basic wind speed

Design wind pressure P_Z can be calculated using the formula,

$$P_Z = 0.6\ V_Z^2$$

Where V_Z = Design wind velocity (m/sec) at z (m) height from the base

P_Z = Design wind pressure (N/m^2) at z (m) height from the base

Table 1 Design wind velocity and design wind pressure value at 10 m, 15 m and 18 m height of the structure

Sl. No.	Height (m)	V_b (m/sec.)	K_1	K_2	K_3	V_z (m/sec)	$P_z = 0.6\ V_z^2$ (N/m^2)
1	10	39	1	1	1	39	912.600
2	15	39	1	1.05	1	40.95	1006.141
3	18	39	1	1.06	1	41.34	1025.397

For wind analysis load combination as per limit state method of design is taken with factor 1.2(DL+LL+WL).

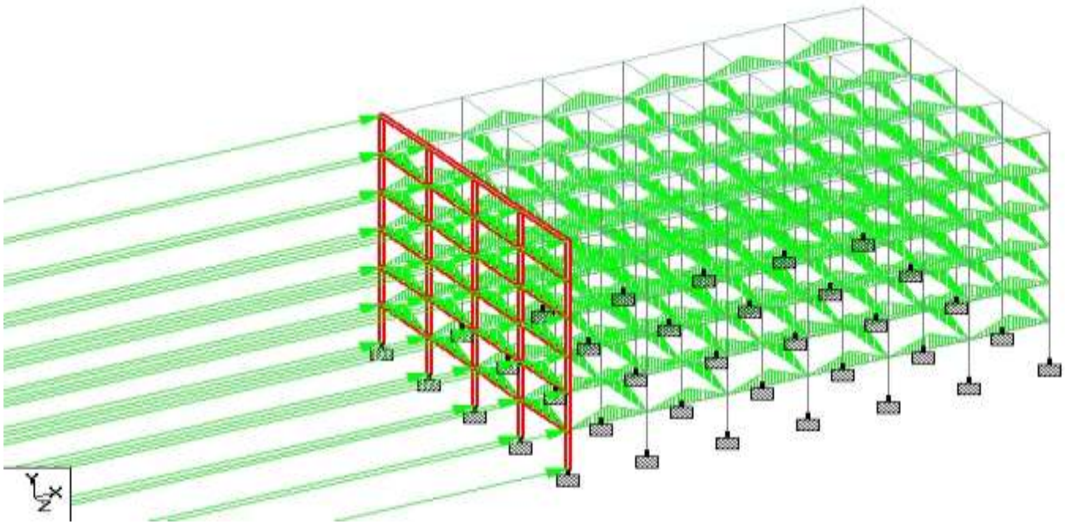

Fig 3. Shows structure after application of wind load

Earthquake load (EL): Earthquake forces constitute of both horizontal and vertical forces on the structure. The vibration caused by the earthquake can be resolved in three mutually perpendicular direction which are two horizontal and one vertical direction. The vertical direction forces do not cause any significant changes in structure but horizontal direction forces at the time of earthquake is considered while designing the structure. **IS 1893-2002** gives the details of calculation of the earthquake forces for the design purpose of the structure. For the seismic design calculation of the structure value of response reduction factor, importance factor of the building, seismic zone selection etc. are considered as per the **IS 1893-2002.** For seismic analysis load combination as per limit state method of design is taken with factor 1.5(DL+LL)**,** 1.2(DL+LL± EL)**,** 1.5(DL±EL), and (0.5DL±1.5EL).

Table 2 Seismic parameter considered in the analysis work

Seismic parameter	Value
Zone factor (Zone II)	0.1
Response reduction factor(SMRF)	5
Importance factor (All general building)	1
Rock/Soil type (Medium soil)	2
Damping ratio	5%

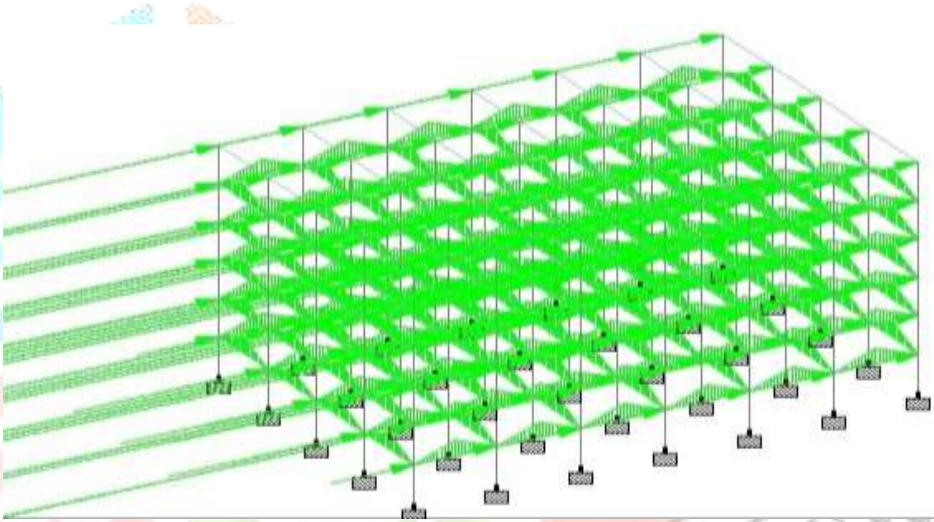

Fig 4. Shows structure after application of seismic load

5. CONCRETE DESIGN

After the completion of analysis of the structure, designing of the structural member such as beam, column and slab is performed by STAAD PRO. For the designing purpose Indian code **IS 456: 2000** is used. Concrete of grade M25 and steel of grade Fe- 415 is taken for design purpose.

Beam: Beam is a structural member which is subjected to compressive force at top fiber and tensile force at bottom fiber when load is applied on it. Generally beam is provided in rectangular shape. The beam used in structure may be of two types.
Singly reinforced beam: when the reinforcement is used only at bottom fiber of the beam the beam is known as singly reinforced beam.
Doubly reinforced beam: when the reinforcement is provided both compression and tension fiber of the beam is known as doubly reinforced beam.

Column: column is a vertical structural member which is subjected to axial compressive force. column transmits the whole structural load to the underneath soil through footing. Column may be circular or rectangular.

Slab: slab is a reinforced concrete structure which are constructed to provide the flat surfaces in the building's floor or roof. The thickness of slab in the structure is taken as 115 mm and cover used 25 mm. slab is supported by the concrete beams by its four side and usually casting is done monolithically.

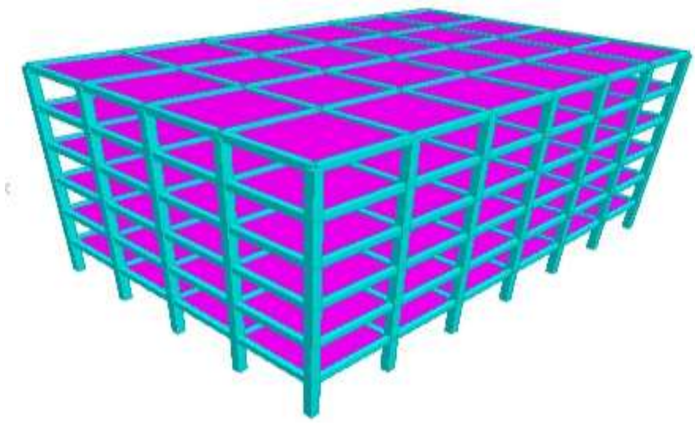

Fig 5. Shows 3D Rendered view of structure

Table 3 Structural member property used in the analysis work

Member	**Size of the member**
Beam	500 mm x 300 mm
Column	450 mm x 450 mm
Slab	115 mm

6. RESULT

After the analysis and design of (G+5) multi-storey framed structure the design detail of beam is obtained with the detail of reinforcement used.

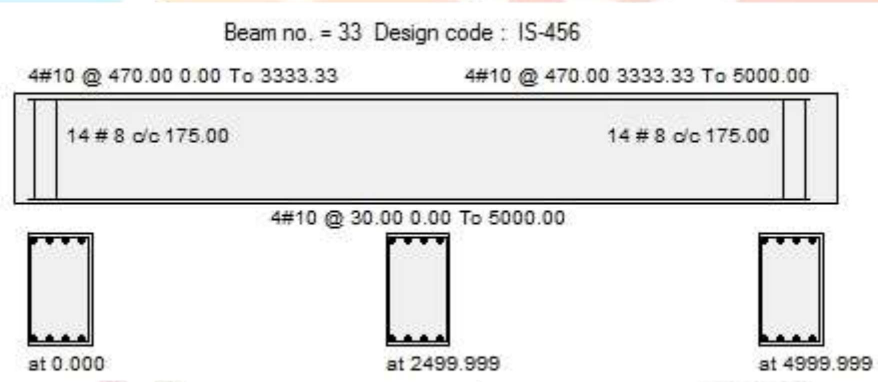

Fig 6. Shows the result obtained for reinforcement detailing in the beam

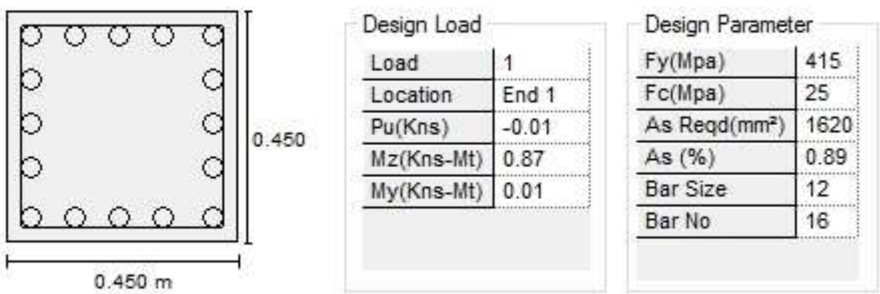

Fig 7. Shows the design parameter and result obtained for reinforcement detailing in the column

Total amount of concrete and reinforcement used for the design of the structure is estimated by the software STAAD PRO.
Total quantity of concrete used: 777.2 cum
Total quantity of steel used: 79576.9 Kg (excluding plate steel)

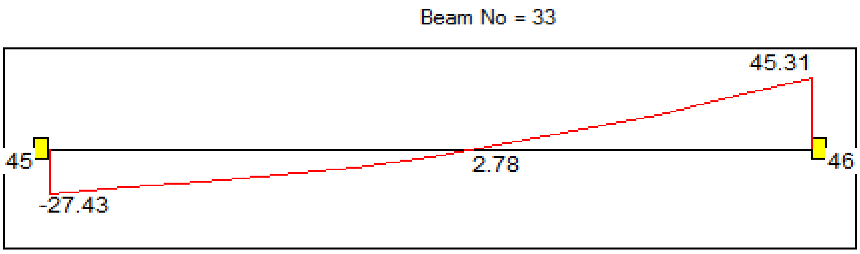

Fig 8. Shows the shear force acting due to combination of load in a beam

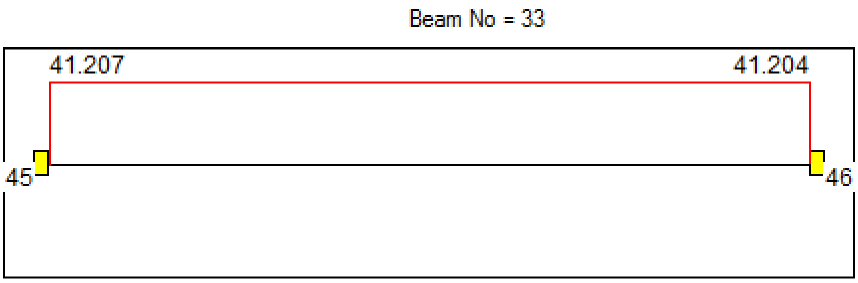

Fig 9. Shows the maximum deflection in a beam

7. CONLUSION

After the analysis and design of G+5 multi storey framed structure using STAAD Pro software the following conclusions obtained.

1. Proposed size of the beam and column can be safely used in the structure.
2. The structure is safe in shear bending and deflection.
3. There is no hazardous effect on the structure due to wind and seismic load on the structure.

REFERENCES

[1] Kulkarni, T., Kulkarni, S., Algur, A., Kolhar, M. H., (2016). Analysis and Design of High Rise Building Frame using STAAD PRO, International journal of research in engineering and technology (IJRET), volume 5, issue 4.

[2] Dutta, M., (2017). Wind Analysis and Design of a Multi-storied Structural Frame Considering using STAAD PRO, International journal of advance in mechanical and civil engineering volume 4, issue 5.

[3] Chaitanya, D. K., Kumar, L. S., (2017). Analysis and Design of a (G+6) Multi-storeyed Residential Building using STAAD PRO, Anveshana's International journal of research in engineering and applied sciences, volume 2, issue 1.

[4] Shivaji, s., Sarvan A., Rama Krishna P., (2017). Analysis of seismic forces for a multi-storied (G+15) residential building by using STAAD PRO, international journal of ethics in engineering and management education, volume 4, issue 2.

[5] IS 875 Part 1 (For Dead load)

[6] IS 875 Part II (For Live load)

[7] IS 875 Part III (For Wind load)

[8] IS 1893-2002 (For Seismic load)

[9] IS 456-2000 (For Concrete Design)